Model Updating of the Ampair Wind Turbine Substructures

Johann Gross, Tobias Oberhardt, Pascal Reuss, Lothar Gaul

Keywords: Dynamic Substructuring, Model Updating, Experimental Modal Analysis

Abstract The modified Ampair 600 is a small size wind turbine which acts as a model to apply substructuring methods in simulation and experiment. The assembly consists of several substructures having very different material properties. Since the parameters of these materials are unknown, model updating is applied to the substructures to obtain validated finite element models. Based on experimentally determined modal parameters the finite element models are adapted to achieve acceptable vibration behavior. Therefore, an objective function consisting of the deviation of the eigenfrequencies and eigenvectors is used to determine Young's modulus, density and Poisson's ratio of each material. In this contribution the finite element models of the substructures are presented. The models are established from CAD models based on manual measurements of the geometry. Identified parameters are given and the resulting correlation is considered.

1 Introduction

Dynamic Substructuring methods offer the possibility to model high order finite element models in an efficient way [1]. A separated representation of the dynamics of the participated substructures enables the application of model reduction methods like the Craig-Bampton method and a later assembly of the different parts by Component Mode Synthesis (CMS). Therefore, the degrees of freedom (DoF) can be reduced drastically. Additionally, a validation of the components can be done individually in a more efficient way. In order to get validated finite element models, model updating methods [2] can be applied to identify material parameters. For example measured modal parameters can be provided to an optimization such that the deviation of the simulation model can be minimized automatically. Within this contribution such a model updating procedure is presented. Starting with the experimental results from Experimental Modal Analyses of the blades in different boundary conditions in Section 2 the results of the rotor assembly are presented. In Section 3 detailed modeling representation of the substructures is given. Adjacently, the definition of the optimization is shown in Section 4. Results of the model updating of the blade are given in Section 5 whereas in Section 6 a short conclusion and information about further investigations is given.

Johann Gross, Tobias Oberhardt, Pascal Reuss, Lothar Gaul
Institute of Applied and Experimental Mechanics, University of Stuttgart, Pfaffenwaldring 9, 70550 Stuttgart
e-mail: {gross,reuss,gaul}@iam.uni-stuttgart.de

2 Experimental results

In order to have reference data for the model updating of the finite element model, three Experimental Modal Analysis (EMA) are done to identify the modal parameters [3]. In the first subsection of this chapter the results of the EMA of the blades in free boundary condition are presented. These results are used to provide the eigenfrequencies and the eigenvector which are further used for the model updating. Subsequently, the results of the blades under a clamped boundary condition at the bolted joints are presented. These measurements verify the validity of the updated finite element model which is adapted to the parameters from the case under free boundary conditions. Finally, the results of the EMA of the assembly consisting of three blades and the hub are shown. For an identification of the material parameters of the hub, these results can be used in the case that the blades are already identified and modeled correctly.

2.1 EMA of the single blades (free free condition)

All three blades are discretized by a grid of 19 measurement points, Fig. 1. For the measurements the high pressure side of the blade is used and considered being a plane surface for simplicity. The extracted eigenvectors from the experimental

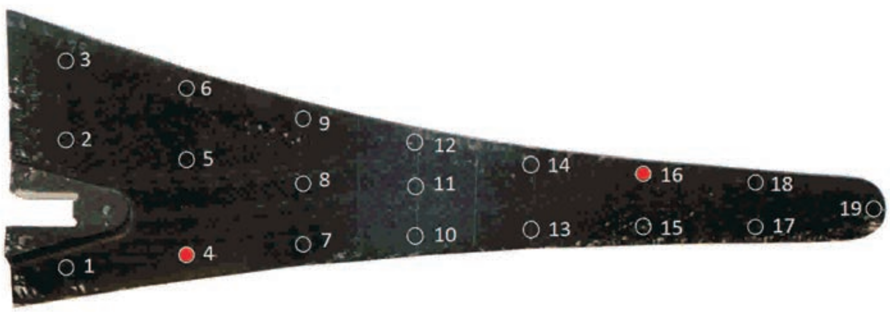

Fig. 1 Blade with 19 measurement points

modal analysis of these 19 points are provided to calculate the MAC values, which are part of the objective function in the model updating optimization. The identified eigenfrequencies for the three blades are listed in Tab. 1. The mode shapes of the first five eigenfrequencies are shown in Fig. 2-4. It can be easily seen, that every blade features different eigenfrequencies, which are induced by diverse material properties and variations in the manufacturing process. It shall be noted that the torsional modes show the highest deviations.

Table 1 Eigenfrequencies of the first five modes for all three blades under free condition.

Mode	Type	Blade 1	Blade 2	Blade 3	max deviation
1	1^{st} *bending*	47,0 Hz	47,7 Hz	47,7 Hz	0,7 Hz
2	2^{nd} *bending*	128,2 Hz	130,3 Hz	130,6 Hz	2,4 Hz
3	1^{st} *torsional*	195,5 Hz	207,0 Hz	206,4 Hz	11,5 Hz
4	3^{rd} *bending*	250,6 Hz	252,9 Hz	251,3 Hz	2,3 Hz
5	2^{nd} *torsional*	329,0 Hz	331,2 Hz	343,6 Hz	14,6 Hz

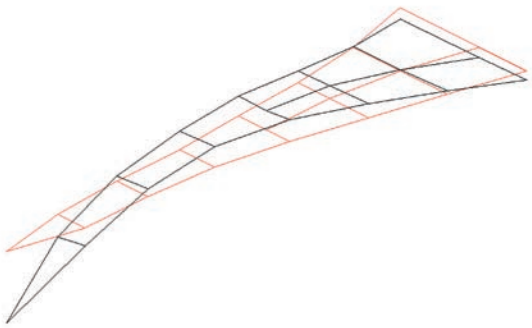
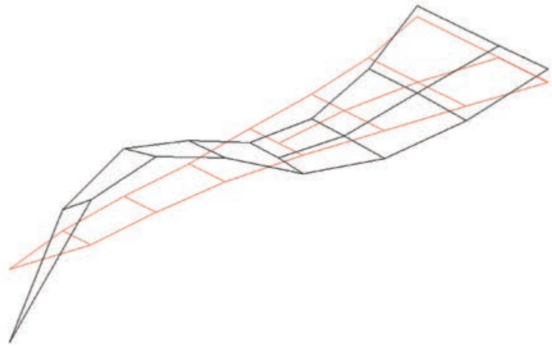

Fig. 2 Left: First bending mode under free boundary condition. Right: Second bending mode under free boundary condition.

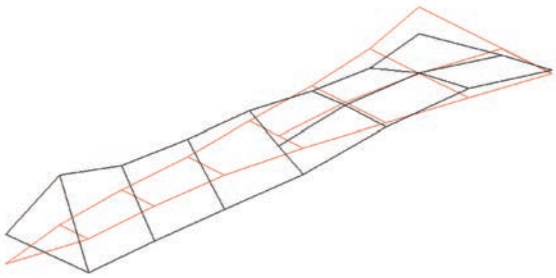
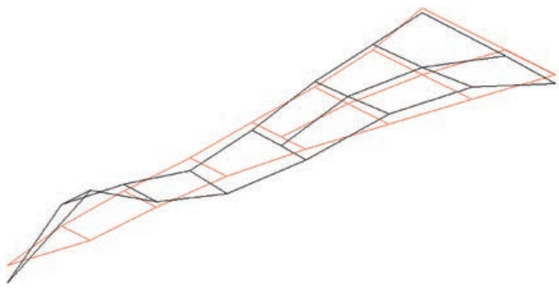

Fig. 3 Left: First torsional mode under free boundary condition. Right: Third bending mode under free boundary condition.

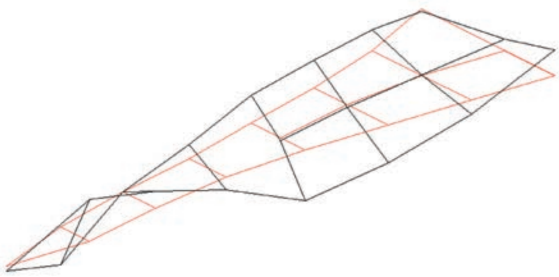

Fig. 4 Second torsional mode under free boundary condition.

2.2 EMA of the single blades (clamped condition)

The modal analysis measurements are repeated on all three blades under a clamped boundary condition. Three screws are used to mount the blade to the table, Fig. 5. In order to avoid contact between the blade and the table counter nuts are used. The results of the EMA are listed in Tab. 2. The mode shapes of the first five modes are illustrated in Fig. 6-8.

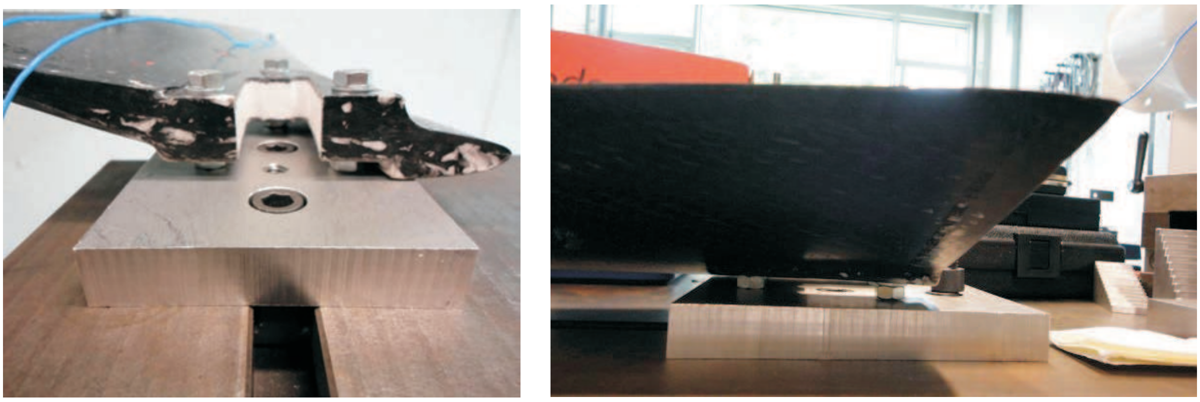

Fig. 5 Fixed boundary condition of the blade

Table 2 Eigenfrequencies of the first five modes for all three blades under clamped condition.

Mode	Type	Blade 1	Blade 2	Blade 3	max deviation
1	1^{st} *bending*	20,0 Hz	20,7 Hz	20,5 Hz	0,7 Hz
2	2^{nd} *bending*	71,1 Hz	70,2 Hz	71,9 Hz	1,7 Hz
3	3^{rd} *bending*	127,7 Hz	137,7 Hz	133,5 Hz	10 Hz
4	4^{th} *bending*	171,5 Hz	179,2 Hz	176,8 Hz	7,7 Hz
5	1^{st} *torsional*	181,1 Hz	190,6 Hz	189,5 Hz	9,5 Hz

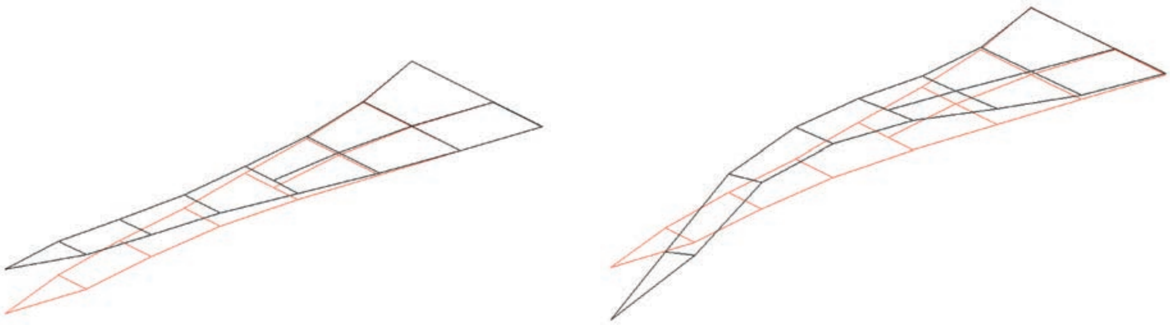

Fig. 6 Left: First bending mode under clamped condition. Right: Second bending mode under clamped condition.

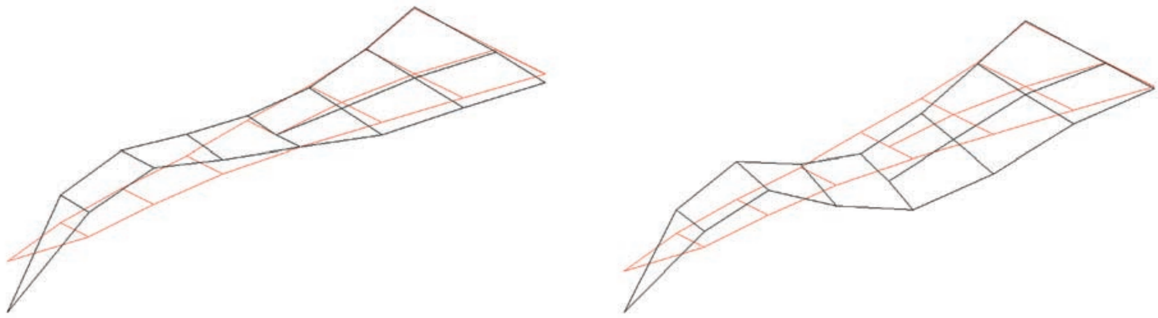

Fig. 7 Left: Third bending mode under clamped condition. Right: Fourth bending mode under clamped condition.

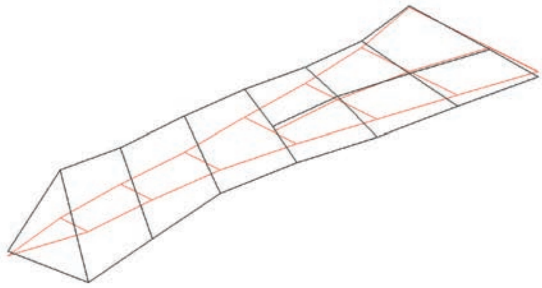

Fig. 8 First torsional mode under clamped condition.

2.3 EMA of the rotor assembly

For a later application of substructuring methods a reference measurement of the rotor assembly consisting of the three blades and the modified hub is established. In a previous step the interior of the hub, Fig. 9 on the right was filled with an epoxy resin to fix the rotational degree of freedom of the blades. The modal analysis is done under free boundary conditions, where the assembly is suspended by a cord with a support frame, Fig. 9 on the left.

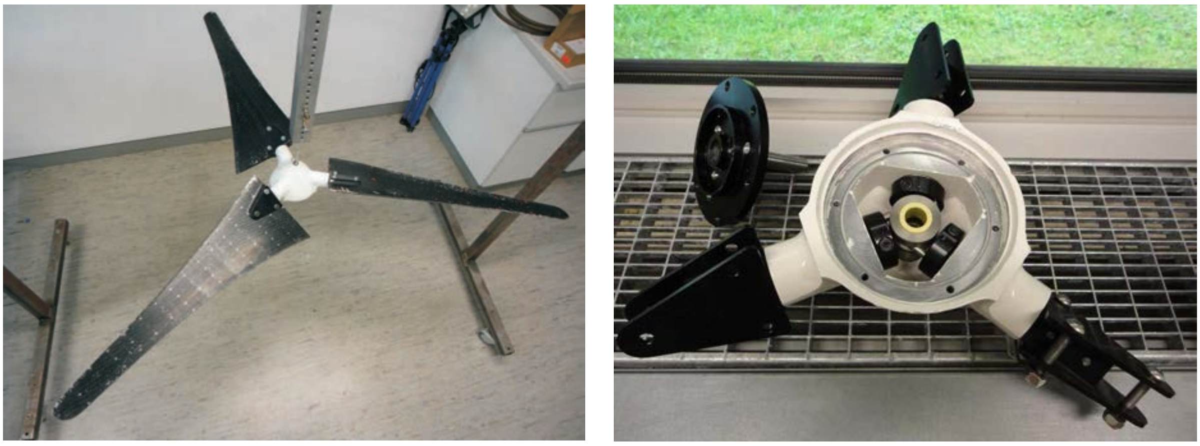

Fig. 9 Left: Rotor assembly in free condition. Right: Interior of the hub.

Mode	Frequency	Unit
1	16,7	Hz
2	23,4	Hz
3	31,6	Hz
4	56,0	Hz
5	75,0	Hz

Fig. 10 Left: Coarse measurement grid for the assembly measurements. Right: First five eigenfrequencies of the assembly.

A more coarse measurement grid is used for this analysis. Five instead of nineteen measurement points are used, Fig. 10 on the left. The determined eigenfrequencies are listed in Fig. 10. The mode shapes of the assembly are shown in Fig. 11-13.

Fig. 11 Left: First mode of the assembly. Right: Second mode of the assembly.

Fig. 12 Left: Third mode of the assembly. Right: Forth mode of the assembly.

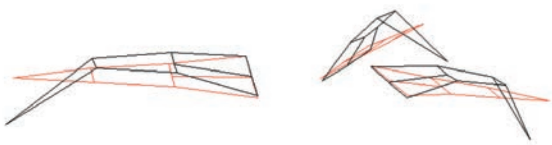

Fig. 13 Fifth mode of the assembly.

Due to the deviations of the material properties between the single blades and within the hub distortion of the cyclic symmetry of the system can be observed. The vibrational energy is not equally spread, but seems rather to be concentrated in single blades, which is indicated by strongly different amplitudes.

3 Modeling of the substructures

The assembly of the wind turbine consists of many different parts having quite different material parameters and are connected to each other in different ways. Since the influence of each individual part on the overall dynamics is unknown all parts are modeled such that individual material parameter can be given to reach best matching between simulation and experiment. The first step toward a finite element model which is able to capture the dynamics of the system is to know the geometry. Therefore, the dimensions of the real parts were recorded manually and converted into CAD models, Fig. 14. Based on the geometry a finite element model of the blade was established. Due to the complicated shape of the blade the

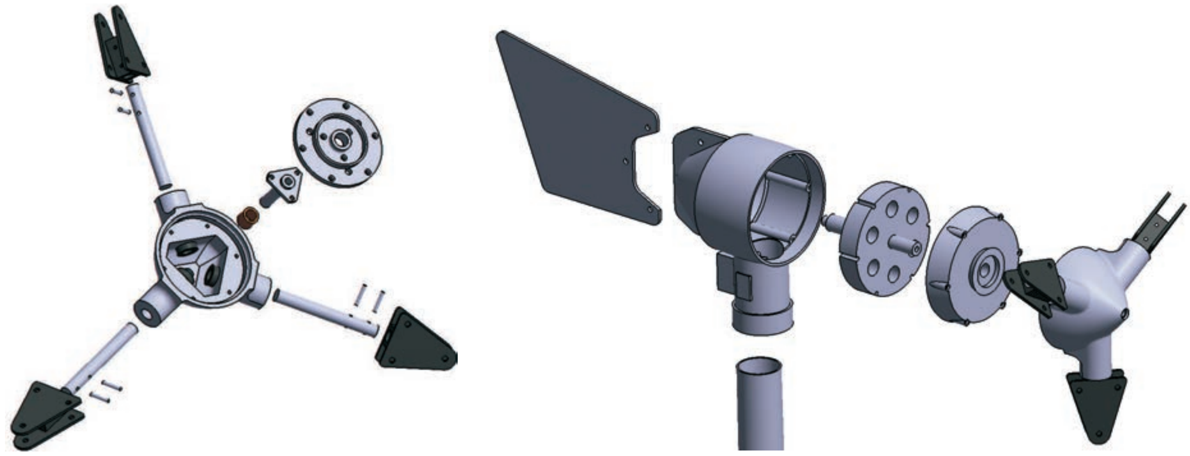

Fig. 14 CAD models of the assembly. Left: Details of the hub model. Right: Turbine assembly parts.

geometry is divided in different sections which can be meshed differently, Fig. 15. The yellow region includes the edges which are sharp-ended. This section is finely meshed. The pink section however is meshed more coarse. The clamping area denoted by the cyan color contains many transitions and is meshed again more accurately, Fig. 16 on the left. In Fig. 15 the nineteen measurement points are additionally indicated in the model by the white dots.

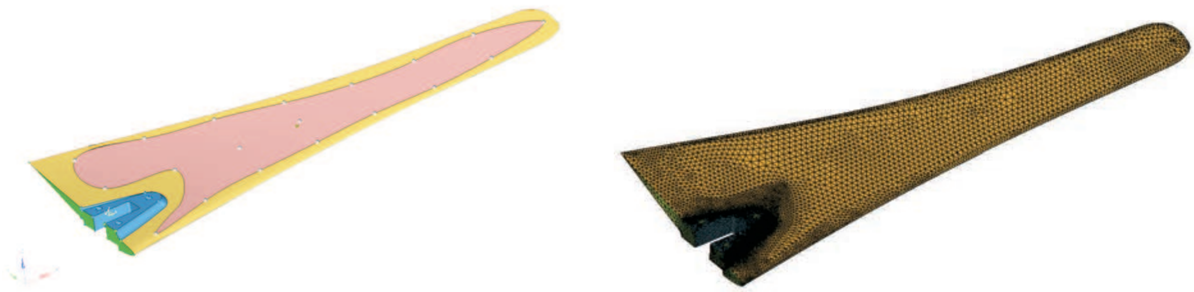

Fig. 15 Left: Geometry model of the blade with measurement points. Right: Finite element model with a fine mesh for the model updating.

Like it can be seen in Fig. 5 the real blade consists of two different materials. The outer layer of the blade is made of carbon fiber reinforced plastic with a thickness of approximately 2 mm. This material is represented in the model by a layer of linear shell elements where a constant thickness of also 2 mm is assumed. The interior of the blade in contrast is made of a structural foam and is represented by linear solid elements. Due to the complex shape of the blade mainly tetrahedron elements are used for the interior. An intersection of the blade is given in Fig. 16 on the right.

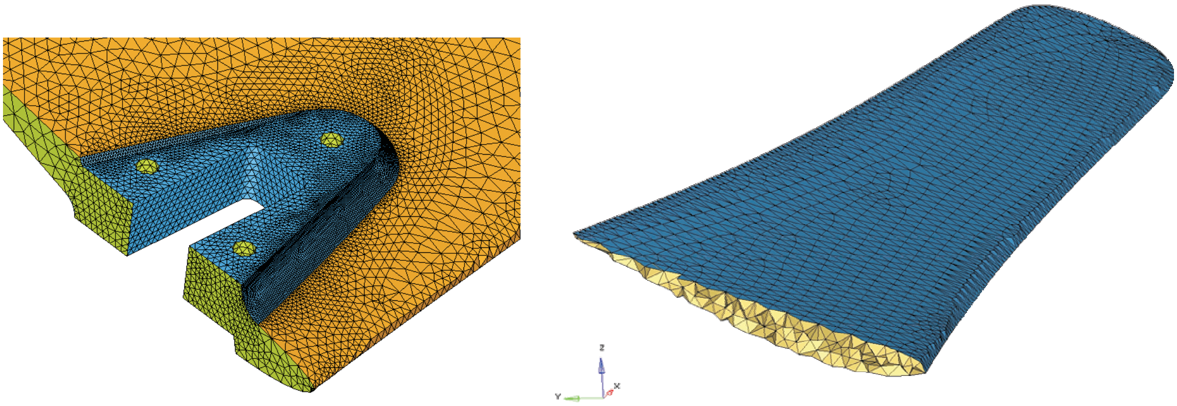

Fig. 16 Left: Zoom on the clamping of the blade. Right: Intersection of the blade. The blue color indicates the shell elements and yellow color the solidelements.

The hub of the wind turbine is a complex part which has numerous components. In Fig. 17 an intersection of the finite element model can be seen. The components are modeled individually and are assembled using compatibility conditions at the contact surfaces. In addition to the parts, which can be seen in the intersection, the epoxy resin is modeled for the sake of completeness. In further investigations measurements of the hub assembly will be established and a model updating will be performed.

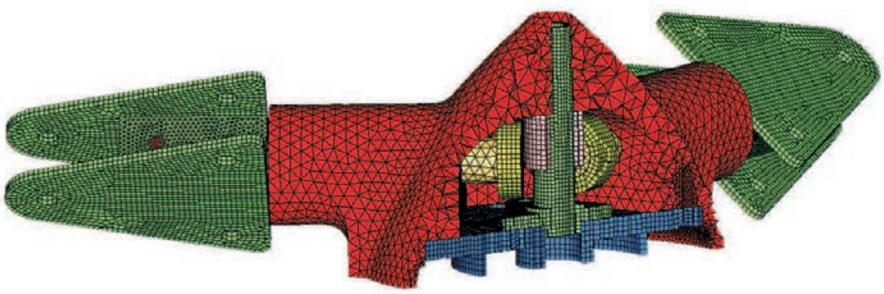

Fig. 17 Mesh of the hub assembly.

4 Model Updating

Model updating is a method to adjust parameters of a simulation model automatically so that it matches the dynamic behavior of the measured part. In the present case the modal parameters from the experiments are used as reference to find material parameters for the finite element model. For this purpose an objective function is created which includes the deviation of the measured and simulated eigenfrequencies and -vectors. Reaching a good result in optimization strongly depends on the quality of the finite element model. Since the finite element models themselves contain uncertainties with respect to the real parts a good agreement for all eigenfrequnecies and -vectors could be unachievable. To counteract this problem a weighted sum is introduced which offers more variability for a good compromise of all considered modes. The weighted sum can be written as

$$\min \sum_{i=1}^{m} w_i f_i(\boldsymbol{x}) = \min(J), \tag{1}$$

where $\boldsymbol{x}$ is the n-dimensional vector of the parameters to be updated, f_i represents the single objective functions, w_i the weighting factors and m the number of the considered objective functions. In the present case the overall objective function J is composed of two functions. One represents the frequencies and the other the eigenvectors such that J can be written as

$$J = \Delta f_{\mathrm{freq}} + \Delta f_{\psi}. \tag{2}$$

The weighted sum of the deviation of the measured and simulated eigenfrequencies is denoted by

$$\Delta f_{\mathrm{freq}} = \sum_{k=1}^{l} w_k \left(\frac{f_{k,sim} - f_{k,meas}}{f_{k,meas}} \right), \tag{3}$$

and the deviation of the eigenvectors in form of weighted MAC values [3] is described by

$$\Delta f_{\psi} = \sum_{j=1}^{n} w_j \left(1 - MAC_j \right) \quad \text{with} \quad MAC_j(\psi_{\mathrm{j,meas}}, \psi_{\mathrm{j,sim}}) = \frac{|\psi_{\mathrm{j,sim}}^T \psi_{\mathrm{j,meas}}|}{(\psi_{\mathrm{j,sim}}^T \psi_{\mathrm{j,sim}})(\psi_{\mathrm{j,meas}}^T \psi_{\mathrm{j,meas}})}. \tag{4}$$

For the optimization the Optimisation Toolbox of MATLAB is used. The finite element model is imported into MATLAB using the Structural Dynamics Toolbox [4] and reassembled in every iteration step. An eigenvalue analysis is operated and the deviations are calculated with the eigenfrequencies and -vectors extracted and imported from the modal analysis.

5 Results

The proposed method is applied to the finite element model of the blade. In a preliminary set up, the optimization algorithm proposes the following set of material parameters, Tab. 3. Using these material parameters the eigenfrequencies listed in Tab. 4 are obtained. The results reveal that for the bending modes a quite good accordance can be reached while the eigenfrequency of the torsional mode shows slight deviation.

Table 3 optimization results for the material parameters.

Parameter	Value	Unit	Parameter	Value	Unit
E_{foam}	500	MPa	E_{carb}	$1 \cdot 10^5$	MPa
ν_{foam}	0.2	-	ν_{carb}	0.5	-
ρ_{foam}	$1 \cdot 10^{-9}$	t/mm^3	ρ_{carb}	$1.5 \cdot 10^{-8}$	t/mm^3

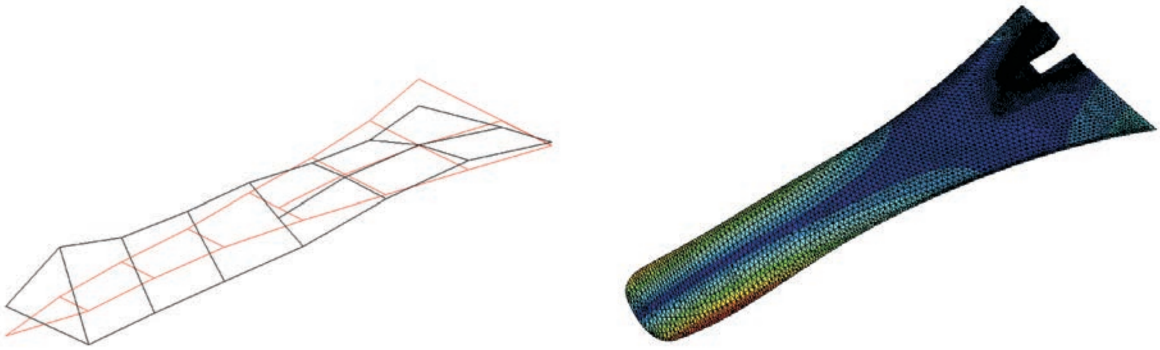

Fig. 18 Left: First measured torsional mode. Right: First simulated torsional mode.

Table 4 Proposed material parameters.

Mode	Type	Frequency	Unit
1	1^{st} *bending*	48.4	Hz
2	2^{nd} *bending*	134.32	Hz
3	1^{st} *torsional*	220.14	Hz
4	3^{rd} *bending*	254.07	Hz

6 Conclusion

Within this contribution a method to identify parameters for the finite element models of the Ampair 600 wind turbine is presented. Therefore, measurements for the blades in free and clamped boundary conditions were established as well as for the hub assembly. The modal parameters were extracted and provided to a model updating routine. The model updating uses a finite element models which were constructed based on the CAD models of the parts. Further effort will be made to identify parameters of the whole assembly. Additional measurements and simulations will be done to be able to apply substructuring methods on the wind turbine.

References

1. De Klerk D., Rixen D., Voormeeren S. (2008) General Framework for Dynamic Substructuring: History, Review and Classification of Techniques. AIAA Journal Vol.46 No.5 (1169-1181) doi: 10.2514/1.33274
2. Friswell M., Mottershead J.E. (1995), Finite Element Model Updating in Structural Dynamics, Springer.
3. Ewins D.J. (2009) Modal Testing: Theory, Practice and Application. Research Studies Press LTD. Hertfordshire, England.
4. Balmes E. (2012), Structural Dynamics Toolbox User's Guide, www.sdtools.com